Geometry Modeling for Automated Finite-Element Analysis of Aircraft Conceptual Design

Wu Li and Jay Robinson
Aeronautics Systems Analysis Branch
NASA Langley Research Center

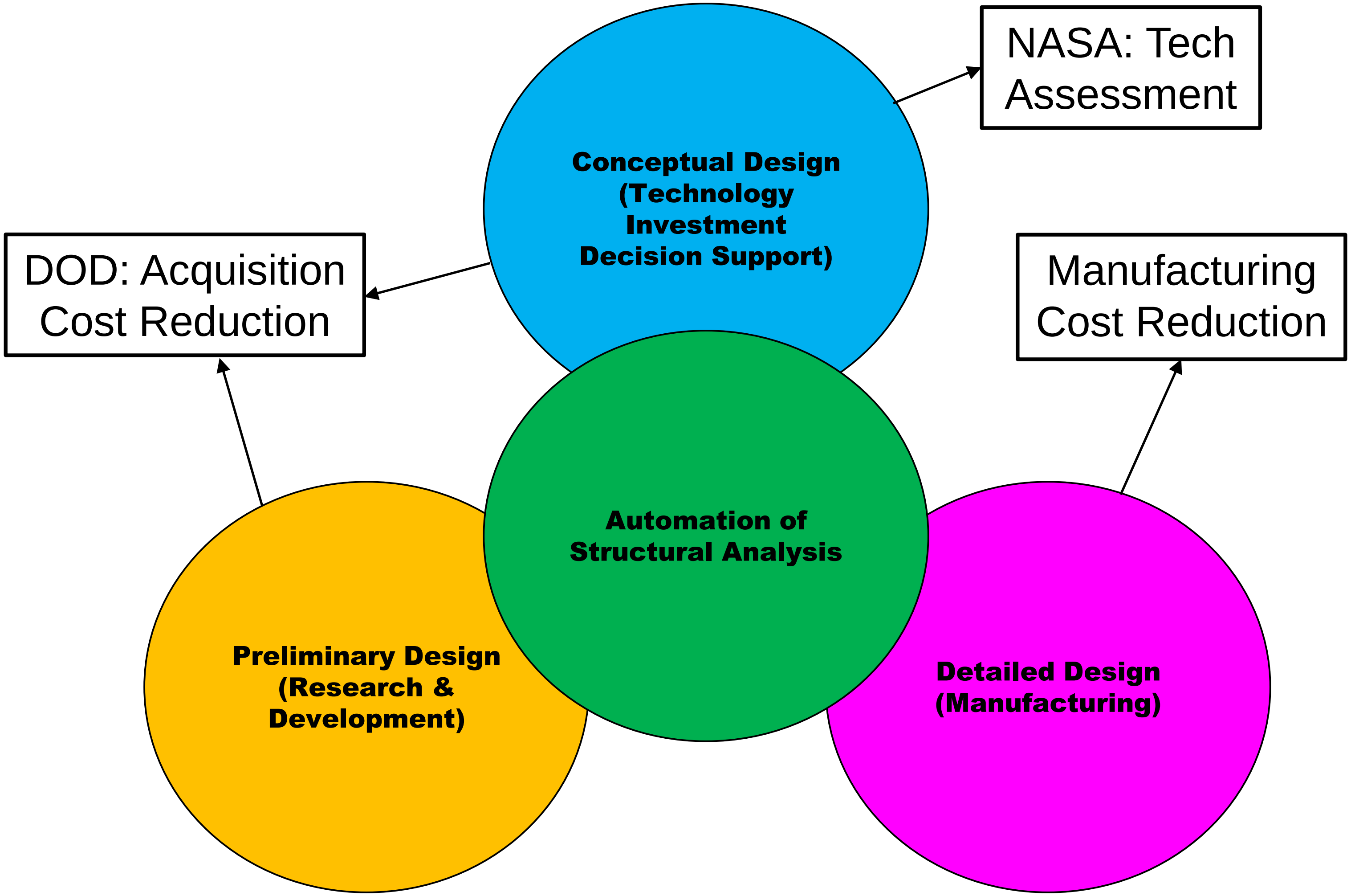
OpenVSP Workshop 2016, August 2016

Outline

- Different automation needs for Finite-Element Analysis (FEA)
- A novel geometry modeling approach for automation of FEA
- Status of current implementation
- Numerical results
- Concluding remarks

Automation of Finite-Element Analysis (FEA)

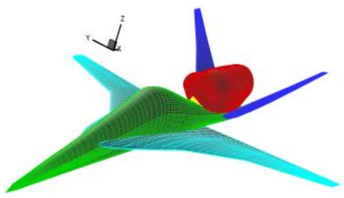
- Conceptual design: Provide more simulation-based info for investment decision support.
- Preliminary design: Enable MDAO studies for interdisciplinary coupling benefits.
- Detailed design: Reduce labor cost for analysis of product mods before manufacturing.



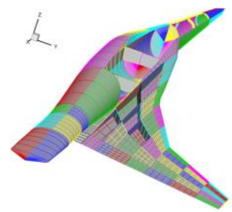
Envisioned Automation Process for FEA

Predefined groups for material property assignment

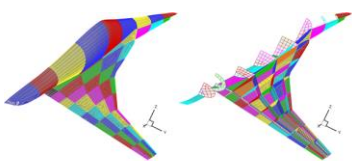
OpenVSP Geometry + Layout Parameter



FEM-Ready Geometry



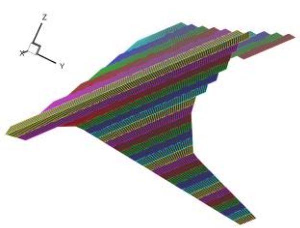
Fully Connected FEM Mesh



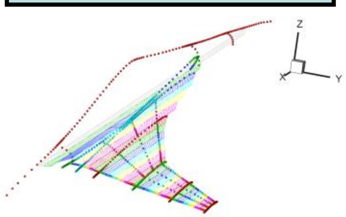
External meshing codes for automated FEM meshing

Automation goal: Problem setup for full vehicle can be completed in minutes instead of hours or days.

Aero Panels



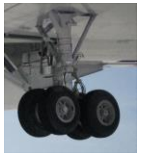
Aero-Structure Coupling



Robust algorithms

Easy-to-use user interface

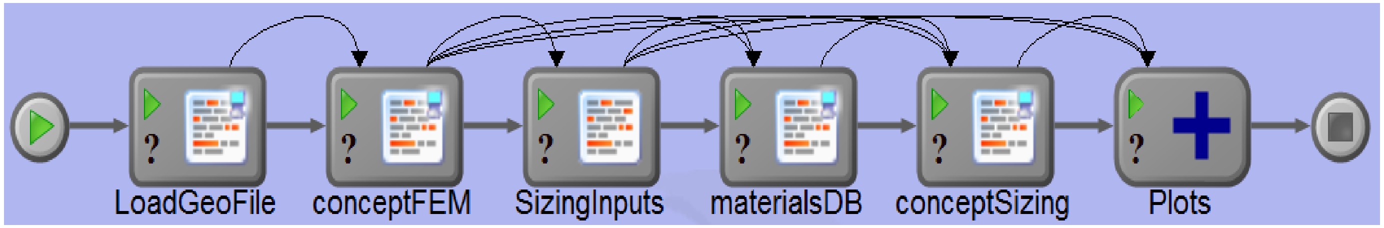
Non-Structural Weights

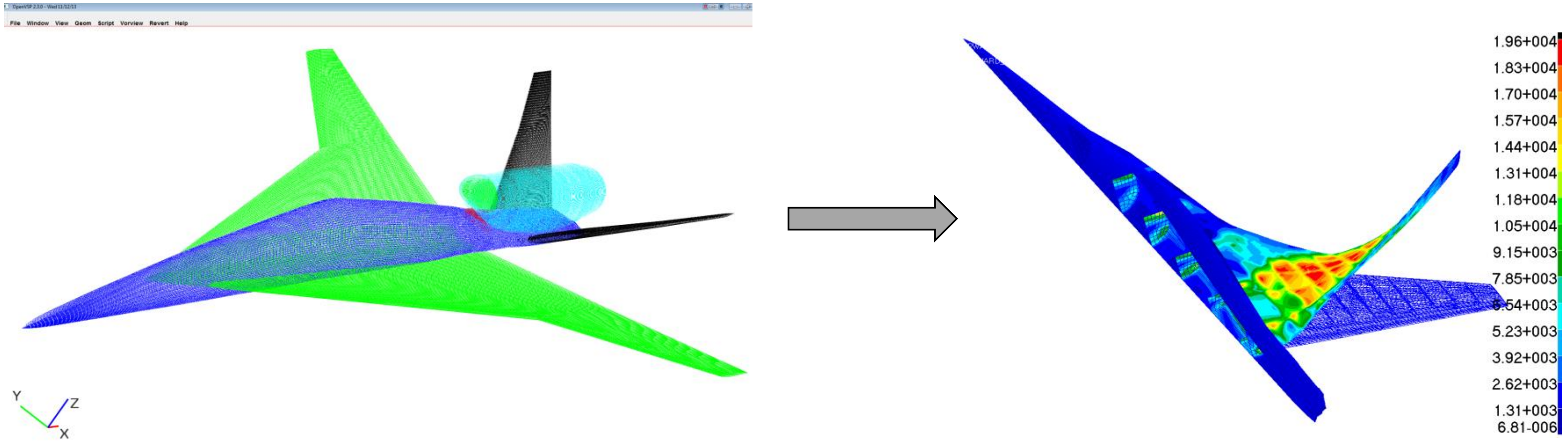


Knowledge-capturing databases for setting up NASTRAN deck

Aeroelastic FEA: Structural Properties, Load/Trim Conditions, Sizing Problem

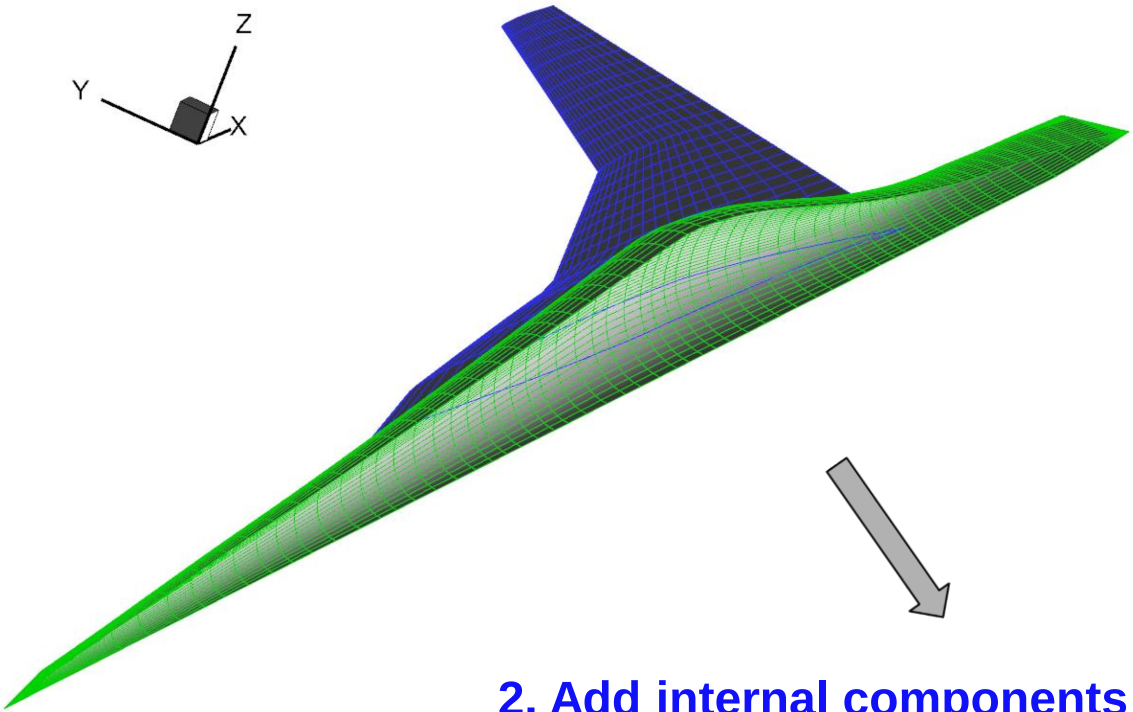
Current Status





- The automated FEA process in ModelCenter starts with an OpenVSP geometry and ends with a NASTRAN 200 solution of any wing-body configuration under two constant pressure load conditions on the wing.
- Two meshing tools: (1) HYBRID mesher in PATRAN, (2) Geompack++
- The automation process can be set up in minutes instead of hours or days.
- Long-term goal: Rapid MDAO capability using static aeroelastic analysis.

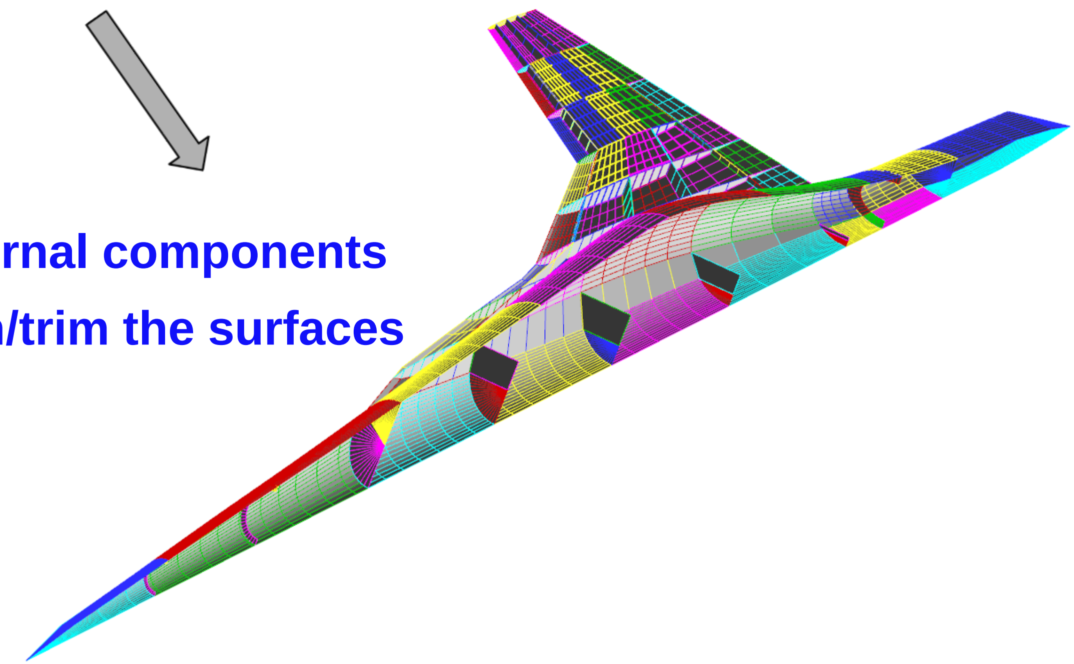
OpenVSP to Structural Geometry



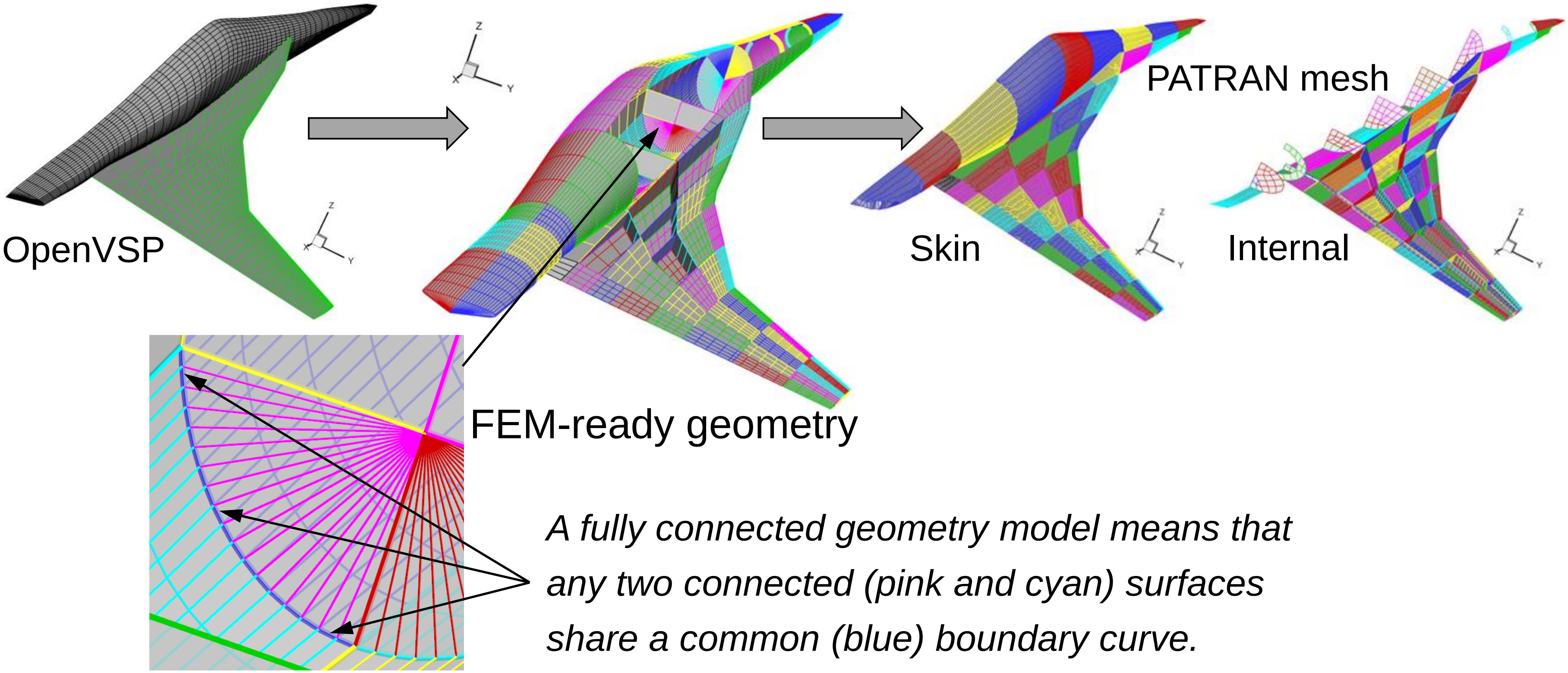
1. Trim OpenVSP Geometry

2. Add internal components

3. Partition/trim the surfaces



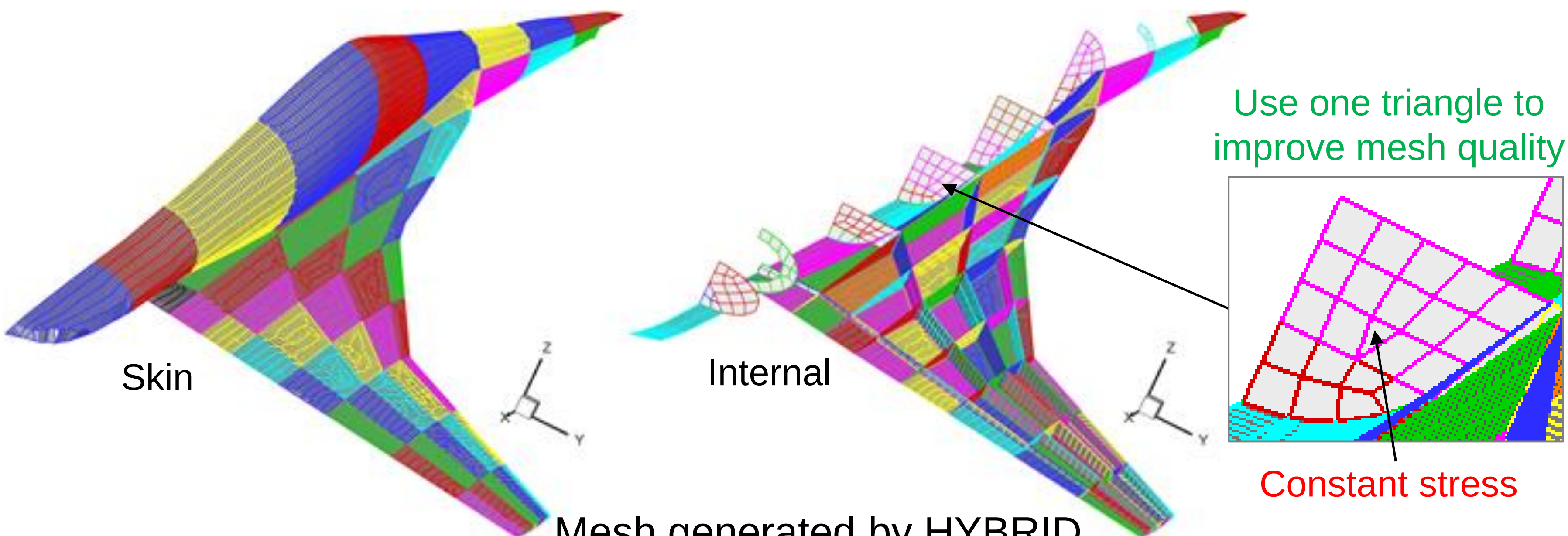
Novelty of FEM-Ready Geometry



- Separation of internal component construction and FEM meshing:
  - Construct a fully connected geometry model for both internal components and skins, which is called FEM-ready geometry.
  - Generate a fully connected finite-element mesh for any FEM-ready geometry without manual preprocessing!
- The separation allows the use of the best available meshing tool in the automation.

HYBRID Mesher in PATRAN

- Export FEM-ready geometry in IGES format as a collection of bilinear B-spline surfaces.
- Use PCL commands in a session file to control the meshing process and export the generated finite-element mesh in a bulk data file.
- The generated mesh is of high quality.
- Number of elements can be controlled by a global length parameter.
- The generated mesh usually contains both quadrilaterals and triangles.

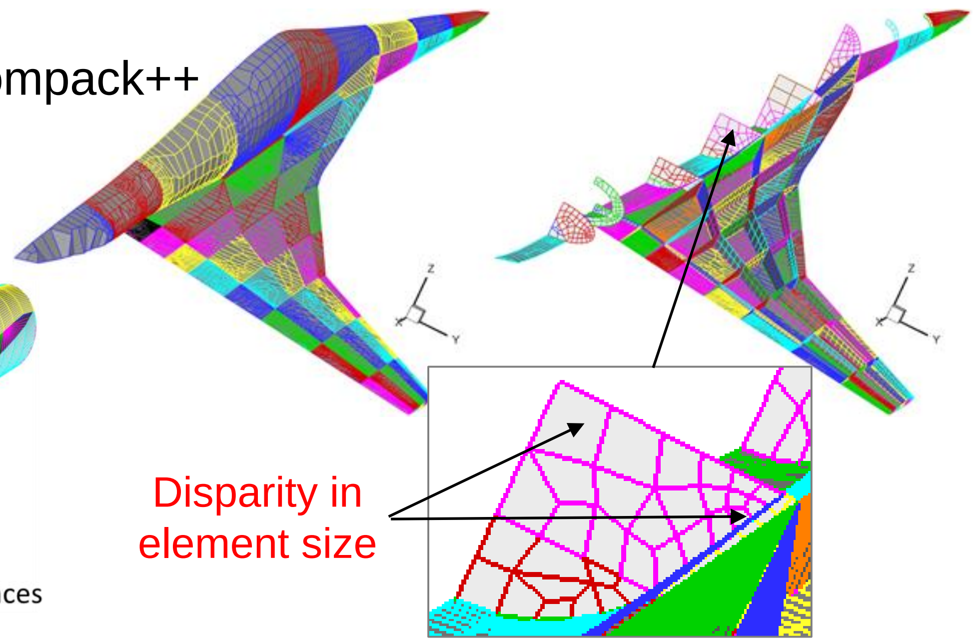


Mesh generated by HYBRID

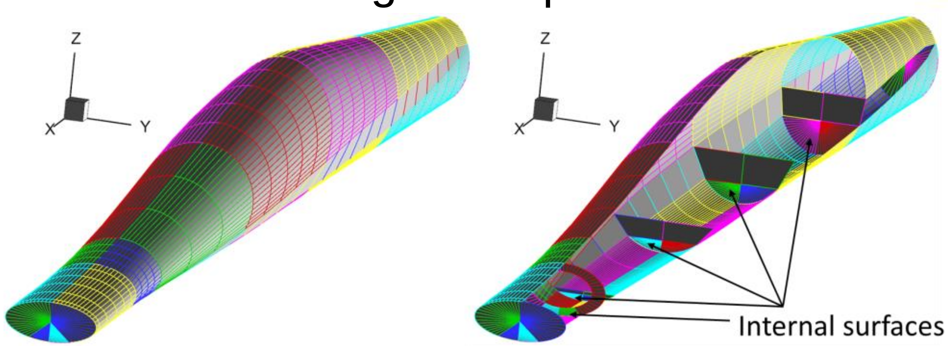
Meshing Software Geompack++

- Geompack++ always generates a quadrilateral mesh for FEM-ready geometry.
- It is not easy to control the mesh size and quality.
- Advancing front method in Geompack++, Barry Joe, Canada.
- Export FEM-ready geometry in the 3D region format required by Geompack++:
    - Each surface is interpolated by a bilinear B-spline surface.
    - All surfaces are rearranged as a collection of watertight compartments.

Mesh generated by Geompack++



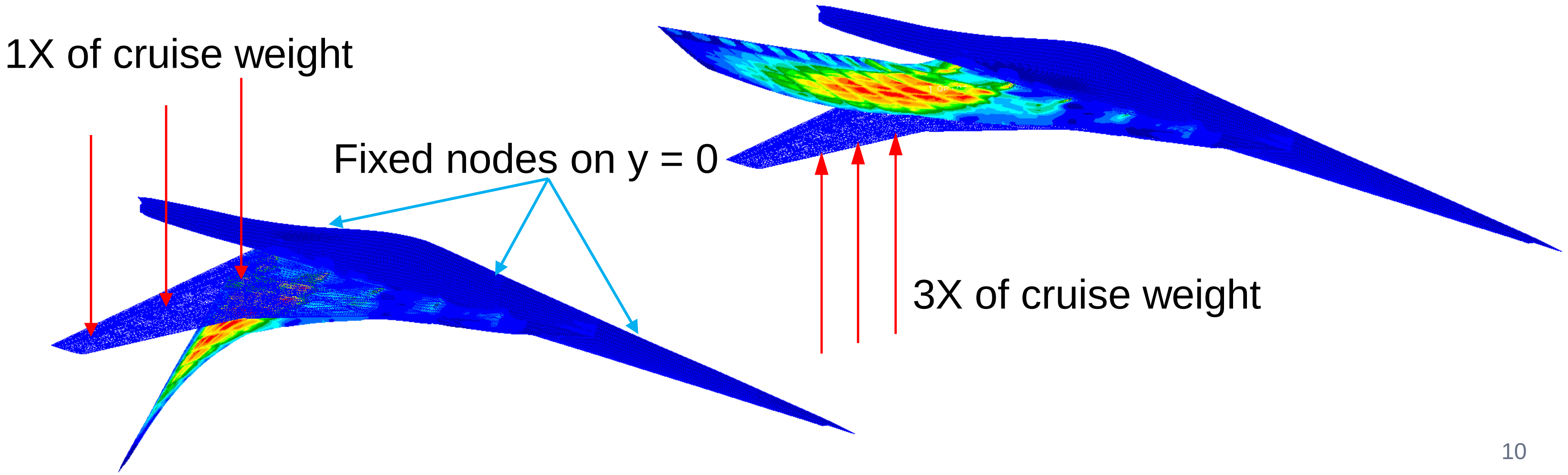
A watertight compartment



A Simple Thickness Optimization Problem

$$\min \ \text{WEIGHT} \ \ \textit{subject to} \ \ |\text{von Mises stress}| \leq \text{stressUB}$$

- All elements for each [bilinear B-spline] surface in the FEM-ready geometry share a thickness design variable.
- Two constant pressure load conditions on the wing:
  - Total wing load in the downward direction is 1X of the cruise weight.
  - Total wing load in the upward direction is 3X of the cruise weight.
- All nodes on the symmetry plane are fixed.
- All elements have the same material property of a generic aluminum alloy.



User Input Requirements (I)

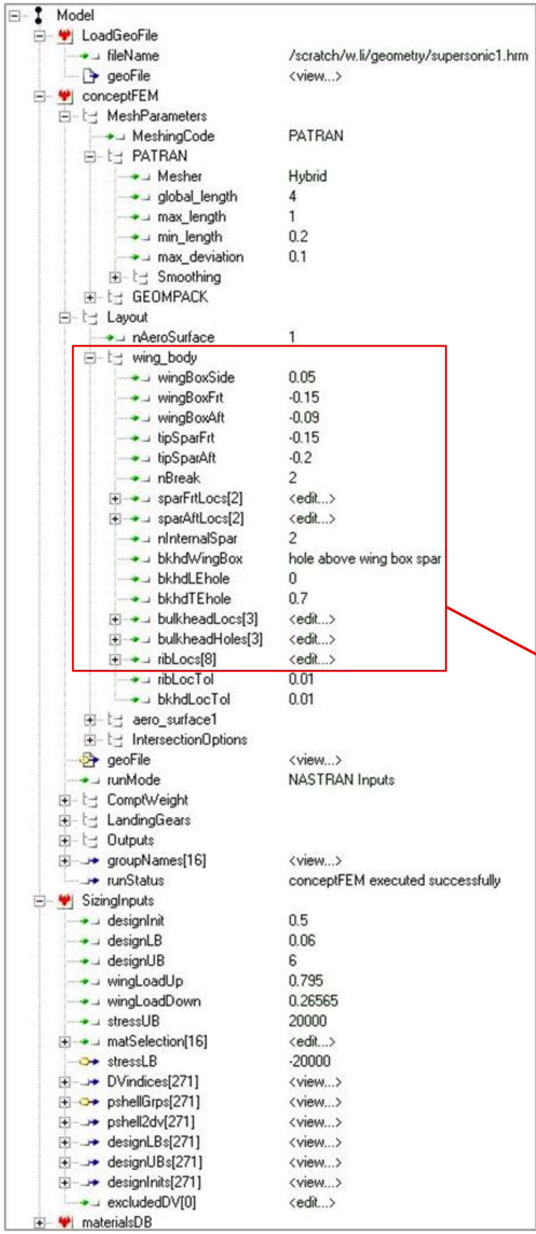
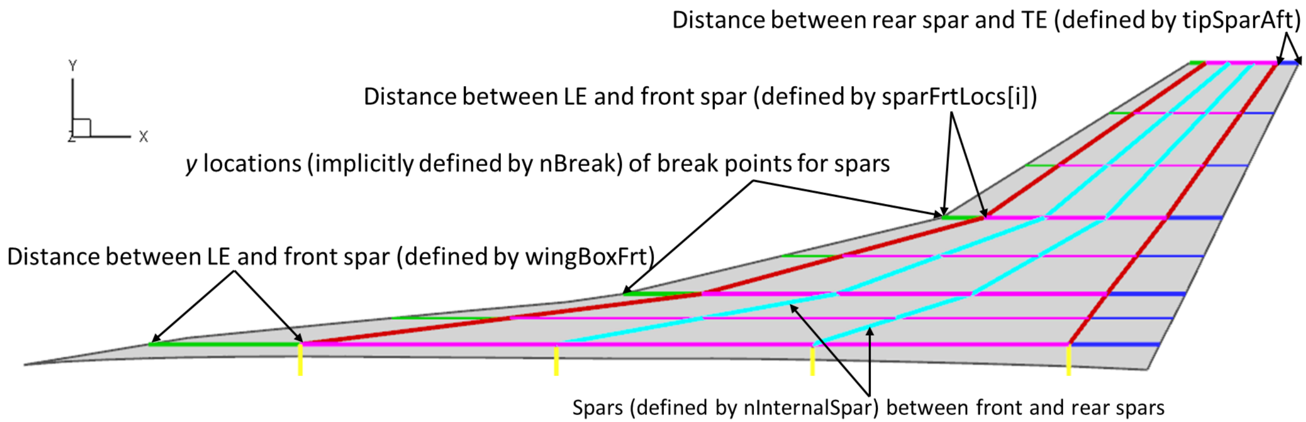
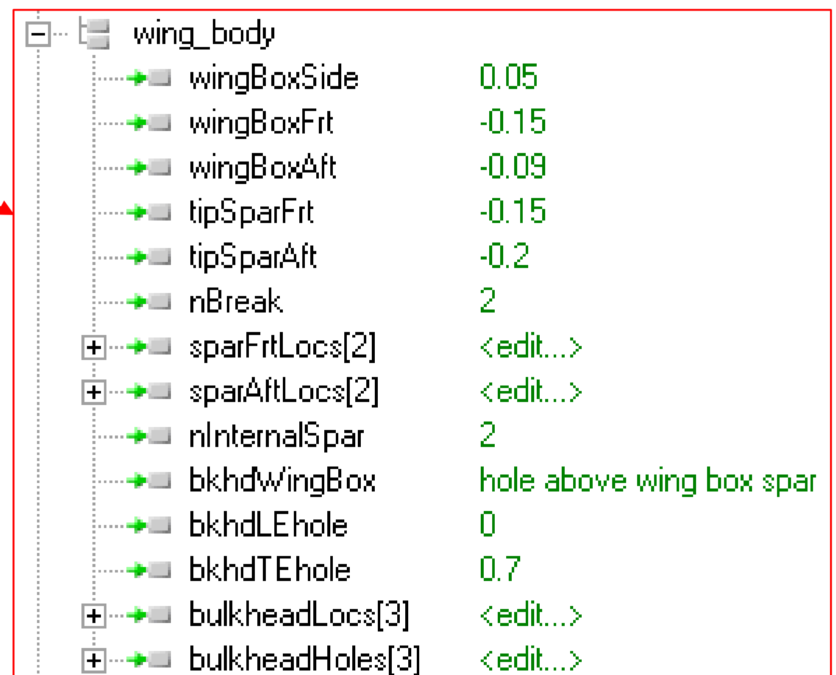


Illustration of some structural layout parameters:



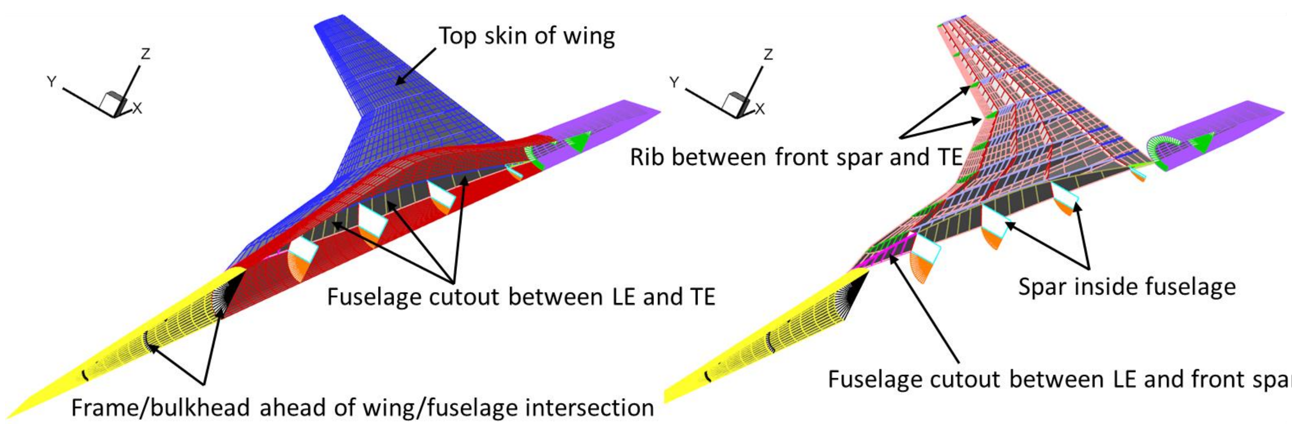


User Input Requirements (II)

Model	
LoadGeoFile	
fileName	/scratch/w.li/geometry/supersonic1.hrm
geoFile	<view...>
conceptFEM	
MeshParameters	
MeshingCode	PATRAN
PATRAN	
Mesher	Hybrid
global_length	4
max_length	1
min_length	0.2
max_deviation	0.1
Smoothing	
GEOMPACK	
Layout	
nAeroSurface	1
wing_body	
wingBoxSide	0.05
wingBoxFrt	-0.15
wingBoxAft	-0.09
tipSparFrt	-0.15
tipSparAft	-0.2
nBreak	2
sparFrtLocs[2]	<edit...>
sparAftLocs[2]	<edit...>
nInternalSpar	2
bkhdWingBox	hole above wing box spar
bkhdLEhole	0
bkhdTEhole	0.7
bulkheadLocs[3]	<edit...>
bulkheadHoles[3]	<edit...>
ribLocs[8]	<edit...>
ribLocTol	0.01
bkhdLocTol	0.01
aero_surface1	
IntersectionOptions	
geoFile	<view...>
runMode	NASTRAN Inputs
CompWeight	
LandingGears	
Outputs	
groupNames[16]	<view...>
runStatus	conceptFEM executed successfully
SizingInputs	
designInit	0.5
designLB	0.06
designUB	6
wingLoadUp	0.795
wingLoadDown	0.26565
stressUB	20000
matSelection[16]	<edit...>
stressLB	-20000
DVindices[271]	<view...>
pshellGrps[271]	<view...>
pshell2dv[271]	<view...>
designLBs[271]	<view...>
designUBs[271]	<view...>
designInits[271]	<view...>
excludedDV[0]	<edit...>
materialsDB	

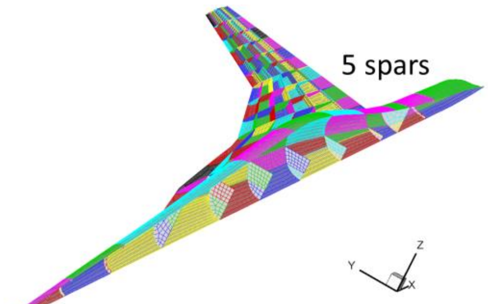
SizingInputs	
designInit	0.5
designLB	0.06
designUB	6
wingLoadUp	0.795
wingLoadDown	0.26565
stressUB	20000
matSelection[16]	<edit...>
stressLB	-20000
DVindices[271]	<view...>
pshellGrps[271]	<view...>
pshell2dv[271]	<view...>
designLBs[271]	<view...>
designUBs[271]	<view...>
designInits[271]	<view...>

Illustration of some predefined groups:

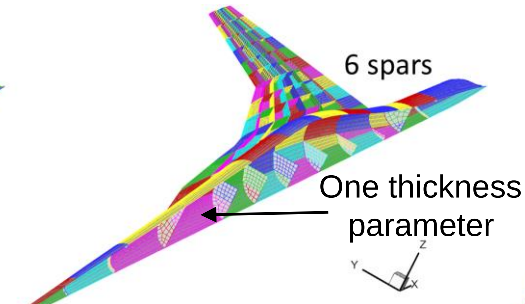


Verification Cases

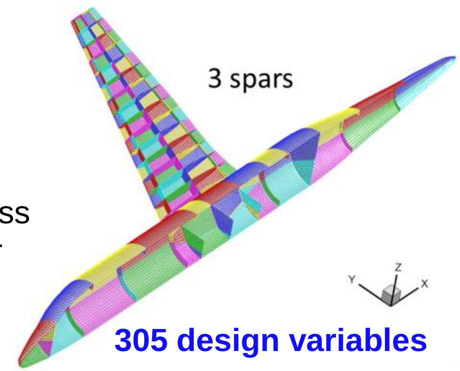
- Four structural layouts for a subsonic business jet and a supersonic low-boom demonstrator concept are used to verify the automation process.
- Five meshes are generated for each layout.
- All elements for one surface in FEM-ready geometry share one thickness design variable.



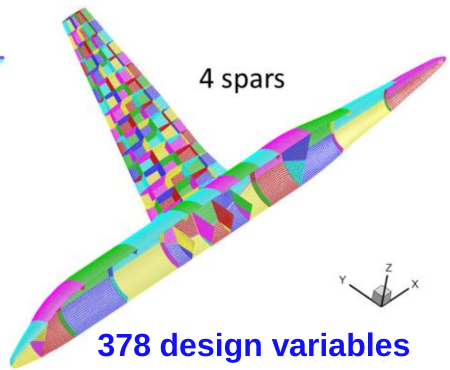
324 design variables



377 design variables

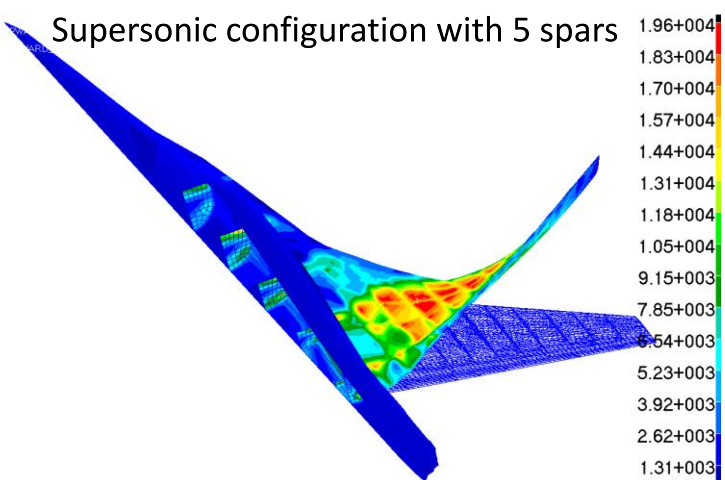


305 design variables

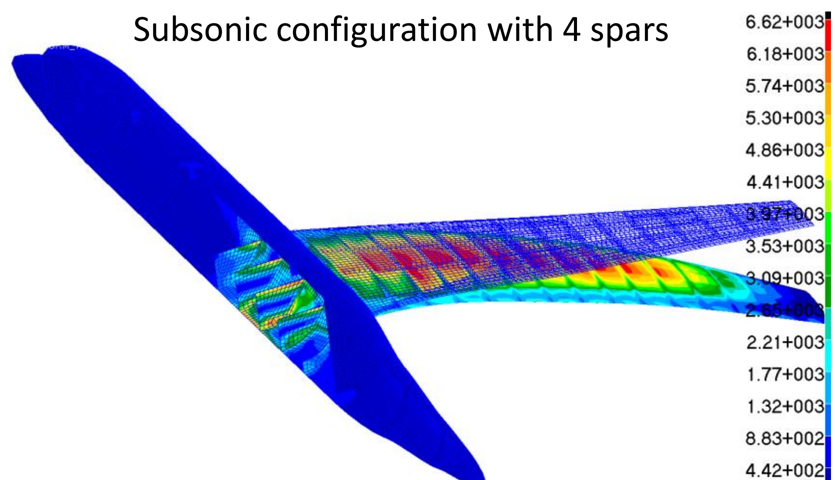


378 design variables

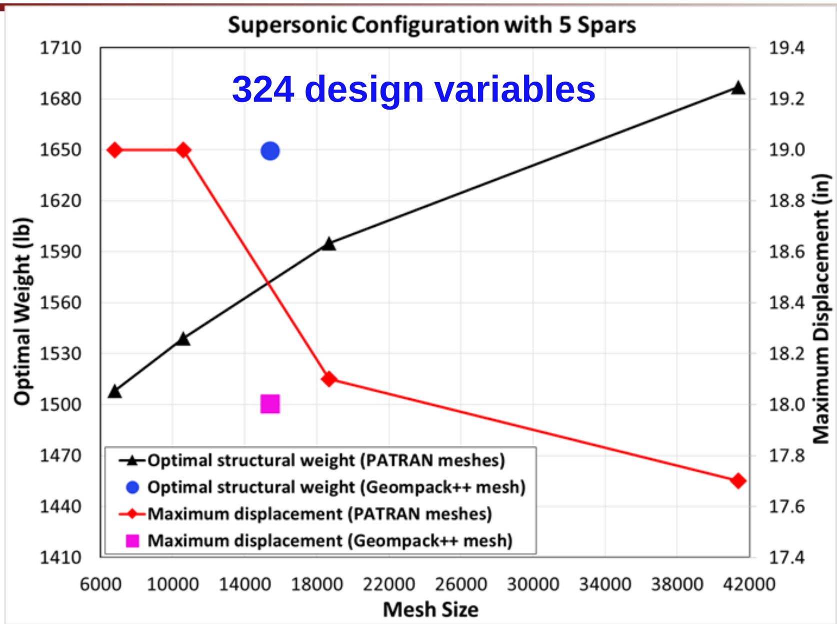
Supersonic configuration with 5 spars

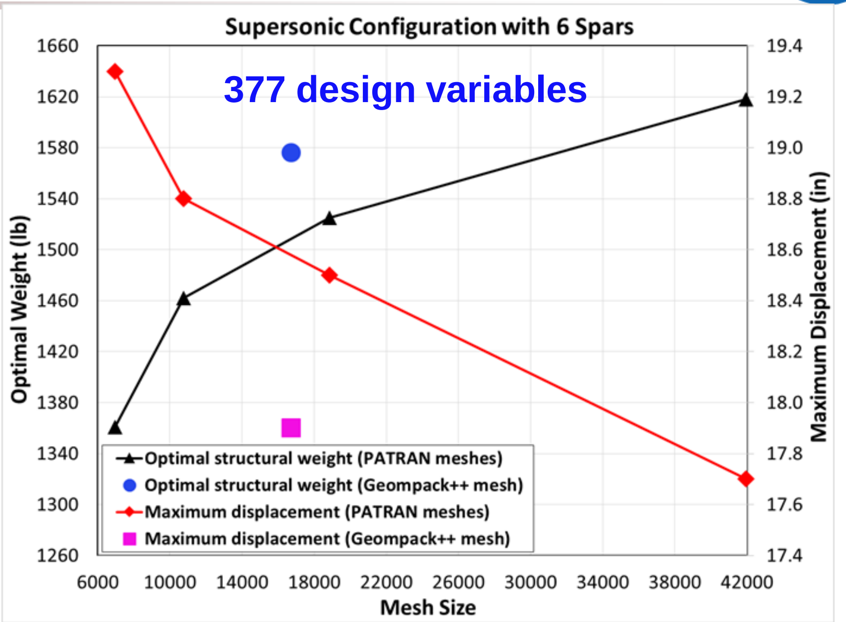


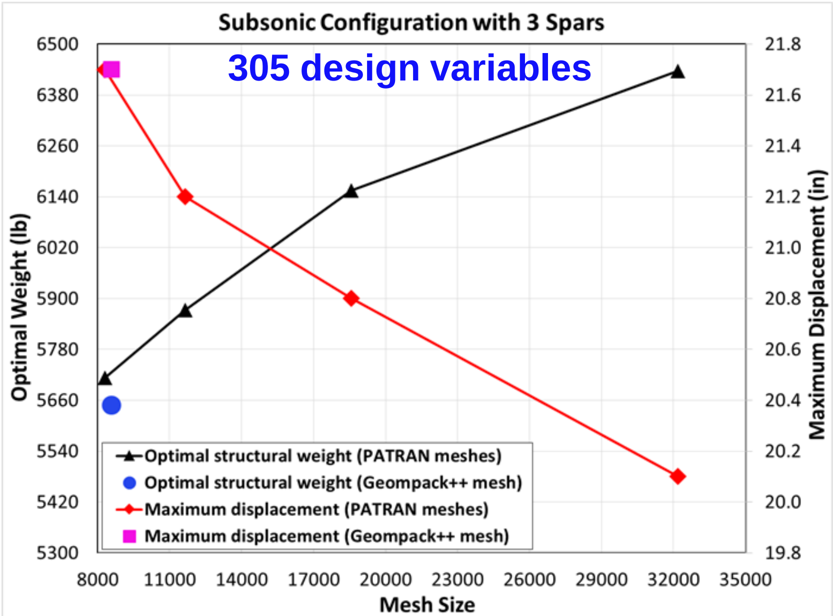
Subsonic configuration with 4 spars

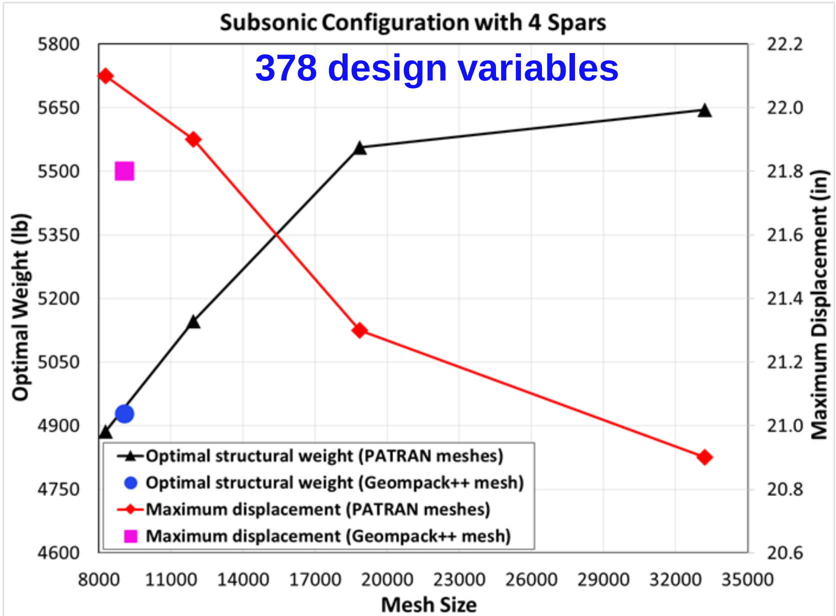


Sensitivity to Mesh Size

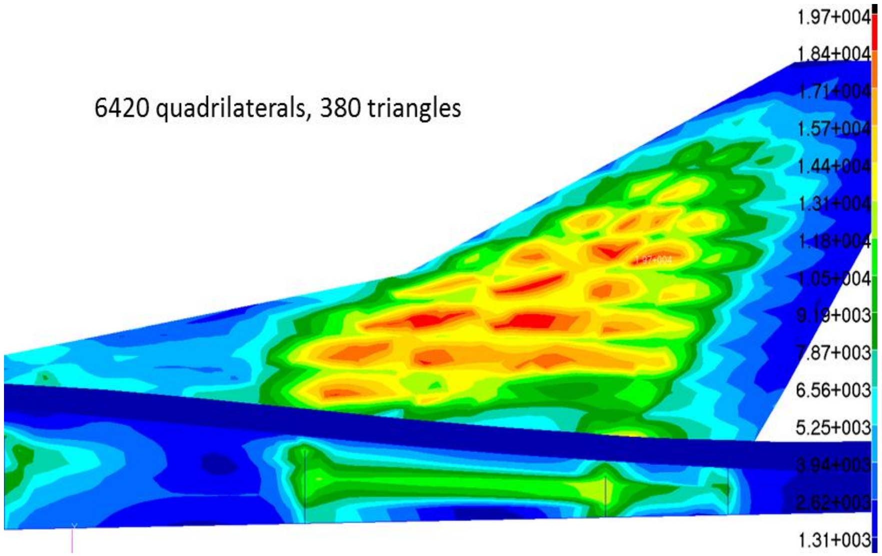


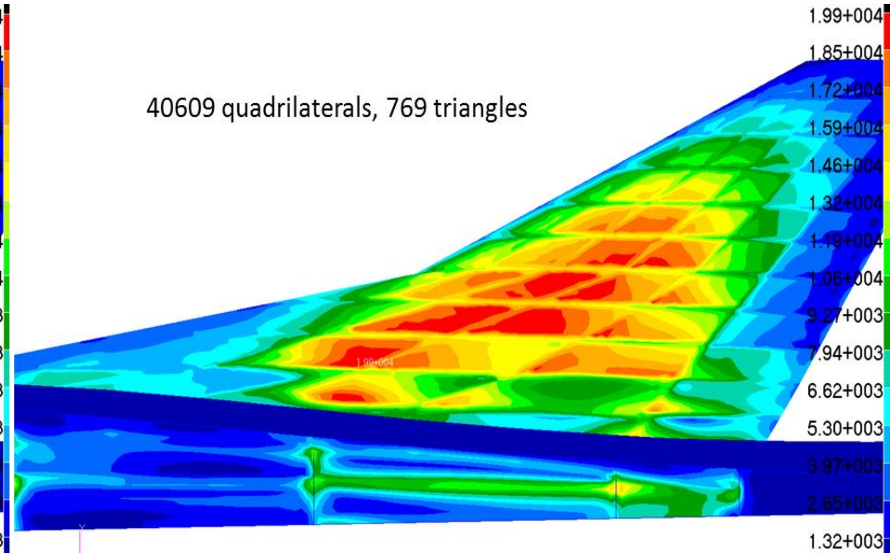


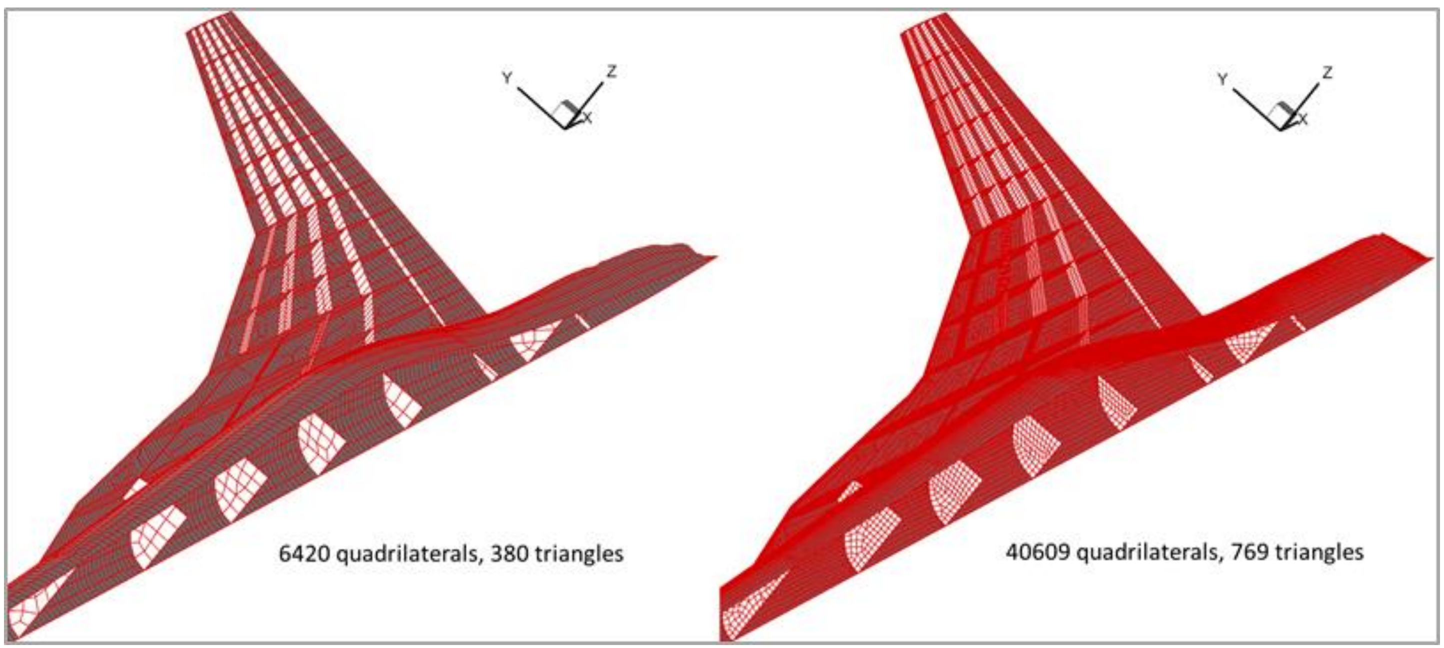




Meshes and Stress Contours

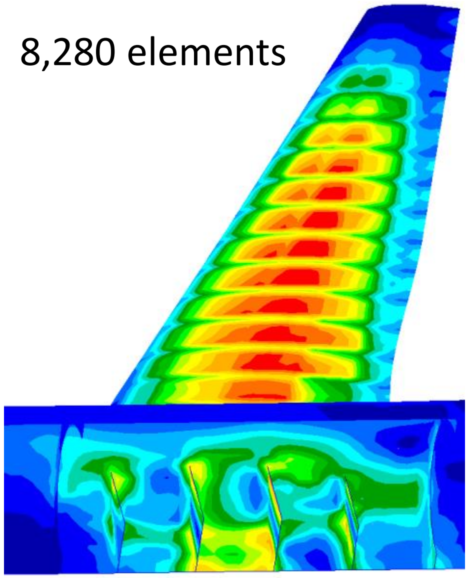


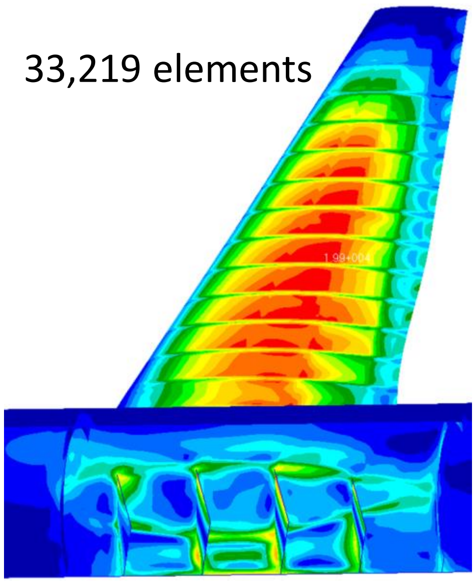




Concluding Remarks

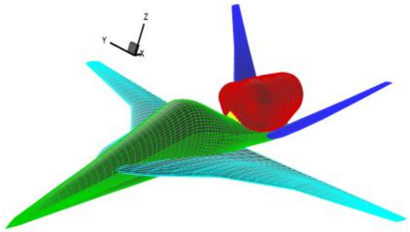
- The automated FEA process starts with an OpenVSP geometry and ends with a NASTRAN 200 solution for thickness optimization.
- The automation process is versatile and robust.
- The automation process can be set up in minutes instead of hours or days.
- The process is verified with 20 meshes for 4 layouts of 2 configurations.
- For each layout, the stress contour plot for a coarse mesh resembles a smeared version of that for a fine mesh.
- Automated mesh generation: (1) FEM-ready geometry for internal components and skins, (2) External meshing tools to generate a fully connected FEM mesh.
- External meshing tools: PATRAN and Geompack++





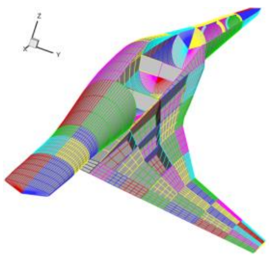
FEA Process for OpenVSP?

OpenVSP Geometry



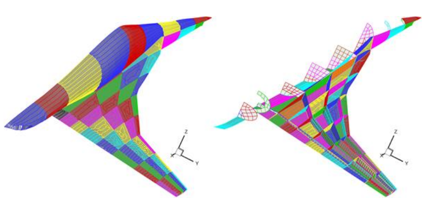
OpenVSP FEM GUI: Top view of FEM layout, automated trim and partition of 3D surfaces

FEM-Ready Geometry



OpenVSP FEA GUI: Structural Properties, Load/Trim Conditions, Sizing Problem

FEA Model for Aeroelastic Analysis



Free FEM meshing tool: Geompack++ for non-commercial use
Tool integration: OpenMDAO or OpenVSP scripts

Questions for Discussion

1. Is it possible to develop a 2D structural layout interface in VSP and the underlying numerical algorithms to generate a FEM-ready geometry model?
2. Could a standard geometry definition requirement be established for automated finite-element meshing?
3. Is a simple finite-element model better than a more detailed finite-element model during conceptual design?
4. What is the best trade-off between the finite-element modeling complexity and the knowledge requirement in setting up the analysis model during conceptual design?
5. Is there a numerical method to merge two structural meshes properly and reliably?
6. Is it time to have a structural weight uncertainty quantification workshop (similar to drag prediction workshop)?

Acknowledgments

- The authors would like to thank Karl Geiselhart at NASA Langley Research Center for providing analysis data of the subsonic business jet.
- The authors would like to thank James Fenbert at Analytical Mechanics Associates for instrumental comments.
- This work is funded by NASA Commercial Supersonic Technology (CST) Project and Transformational Tools & Technologies (T^3) Project.

Backup (I): Approaches for Automated FEA

- Component-Level FEM Followed by Merging
  - FEM mesh intersection (M4 and GeoMach [UMich])
  - NASTRAN glue operation (Sharon Padula's LOFT and Jesse Quinland's modification of HCDstruct for D8)

- Defined Intersection and Component Scripting
  - Applicable to a specific type of vehicles (Jay Robinson's approach for supersonic bizjet, Jesse and Frank's approach for HWB, University of Michigan's approach for strut-braced wing, DLR's detailed wing model and UAV model, and most approaches including some commercial ones)

- Component-Level Trim and Merging Followed by FEM Meshing
  - FEM mesh for a trimmed and partitioned geometry model (ConceptFEM)

Backup (II): Complexity of Automated FEA